A twodimensional quantitative parametric investigation of simplified surface imperfections on the aerodynamic characteristics of a NACA 633418 airfoil

Kruse, Emil Krog; Sørensen, Niels N.; Bak, Christian

Published in:
Wind Energy

Link to article, DOI:
10.1002/we.2573

Publication date:
2021

Document Version
Publisher's PDF, also known as Version of record

Link back to DTU Orbit

Citation (APA):

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.
A two-dimensional quantitative parametric investigation of simplified surface imperfections on the aerodynamic characteristics of a NACA 633-418 airfoil

Emil Krog Kruse1,2 | Niels N. Sørensen2 | Christian Bak2

1 Power Curve ApS, R&D, Aalborg, Denmark
2 Department of Wind Energy, Technical University of Denmark, Roskilde, Denmark

Correspondence
Email: e@cvibe.dk

Funding information
Energy Technology Development and Demonstration Program (EUDP), Grant/Award Number: 64015-0046

Abstract
The aerodynamic performance of a NACA 633-418 airfoil has been analyzed with disturbances in approximately 1000 different configurations focused on the frontal 10% of the airfoil. The configuration parameters are based on field test samples and rain erosion test specimens. The most important trends are presented by 500 configurations each simulated for 6°, 8°, and 10° angle of attack. The simulations are performed with the DTU Wind Energy in-house 2D CFD Reynolds-averaged Navier–Stokes solver, EllipSys2D, combined with the eN transition model for the laminar-turbulent boundary layer transition. The configurations are modeled by a direct geometrical modification of the airfoil shape. The results show that the most important parameters are the position and the depth/height of the disturbance, with up to 35% lift reduction and 90% lift/drag reduction within the specified angle of attacks and disturbance parameter ranges.

KEYWORDS
airfoil, leading edge roughness, LER, NACA 633-418

1 | INTRODUCTION

The global ambition for increasing renewable energy has led to a high demand for wind turbines. As the wind turbines continue to grow in size and number, the focus on optimization and performance for every part of the wind turbine is essential.

With an objective of performance and optimization, one critical part of a wind turbine are the blades. If a wind turbine blade deviates from the design shape, the result might be reduced power production.1,2 A deviation in shape can originate from the production imperfections, wear, and tear or surface contamination such as ice or dirt. These factors are often mentioned in the literature and industry as erosion or roughness, with roughness being a more general term. The performance of a wind turbine blade is most sensitive to roughness close to the leading edge of the airfoil,3,4 giving basis for the general term leading edge roughness (LER).

Prediction of the influence of LER on the airfoil characteristics is useful in both the design phase and cost-benefit analysis of a correction or repair. So far, no proven general method for calculating the influence from LER has been published. Some research has been conducted on ice accretion in the past, due to problems with ice on airplane wings. Ice accretion is often extensive and can extend the chord...
length with orders of magnitude compared to roughness from dirt, erosion, or imperfections.\textsuperscript{5-10} To the authors’ knowledge, no public parametric study exists for LER. However, an attempt to quantify how big imperfections need to be for the airfoil characteristics to deviate from the ideal smooth conditions has been made in Bak et al.\textsuperscript{11} The LER in wind tunnel tests and simulations are often limited to a few erosion patterns, turbulators such as stall strips or zigzag tape or sand grain paper, with in depth studies of either boundary layers or general airfoil characteristics such as lift and drag.\textsuperscript{12-18}

This paper is a part of the Danish Energy Agency (EUDP), the “LER Project” carried out by the Technical University of Denmark (DTU), Aalborg University (AAU), Danish Fundamental Metrology (DFM), and Power Curve ApS. One of the main tasks in the project is research on methods to simulate and predict the influence of LER on wind turbine blades. During this research, it has become clear that LER depends on many factors such as shape, extension, airfoil family and thickness, and Reynolds number.

The focus of this paper is to investigate the important parameters of the shape and extension of LER. This work is not a final solution or overview, but just one step into an overview of aerodynamic impact from LER. The computational fluid dynamics (CFD) code EllipSys2D,\textsuperscript{19,20} is used to calculate two-dimensional airflow around an airfoil and estimate the changes in lift and drag. The calculations have been validated in previous work by the same authors in Kruse et al.,\textsuperscript{21} and this paper is based on the same NACA 633-418 airfoil, given the same conditions, described in Section 2. In the same project, another paper has been made on the subject of substituting the LER, in this case sandpaper and zigzag tape, with a boundary layer model instead of changes in the geometry. These two papers have been published simultaneously.

2 | METHODS

To investigate the aerodynamic effect of shape and extension of the LER, a series of test subjects and photo samples have been collected. Common to most samples are that the LER can be divided into and area with sandpaper-like roughness and an edge between the LER and the undisturbed/smooth surface (from here on noted by “Smooth”). This is illustrated in Figures 1 and 2, where the rough parts is marked with a yellow curve and the edges are marked by arrows.

It can also be seen from Figures 1–3 that LER is highly three dimensional. Since the shape and extension varies with spanwise position of the blade, it has been decided to divide LER into 2D spanwise sections, based on the results from Kruse et al.,\textsuperscript{21} that suggests that 2D polars are integrated over a small span to mimic the flow that basically is influenced by 3D flow because 2D computations are significantly less complex and requires a smaller computational effort.

This study is limited to the edges marked in Figures 1 and 2. Investigations suggests that the edges are the main contributor to changes in aerodynamic characteristics, when a combination of roughness and edges is seen. To achieve a manageable size of the parameter space, the definition of the roughness patch is limited to the chordwise position and the slope of the edges along with the depth/height of the patch. In this definition, a case of LER can only have one height or one depth, which limits the parameters in this study to five parameters. The parameters are illustrated in Figure 4.

The NACA 633-418 airfoil can be seen in Figure 4A. A zoom on leading edge can be seen in Figure 4B. When moving clockwise from the trailing edge, around the airfoil, the first edge encountered is defined to be “Edge 1” whereas the clockwise latter edge is defined as “Edge 2.” A zoom on some Edge 1 examples can be seen in Figure 4C illustrated with four cases compared to the smooth case. The cases are denoted in the

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure1.png}
\caption{Decommissioned blade from 3-MW offshore wind turbine. The blade has been in service for 3 years [Colour figure can be viewed at wileyonlinelibrary.com]}
\end{figure}
legend with comma separated parameters, where the first one is the slope, the second one is the chordwise position, and the latter one being the depth/height. Height is noted with a positive scalar where depth is noted with a negative scalar.

Figure 4D shows an example of a cavity. When the disturbance is either on the suction or the pressure side, the notation exemplified in the legend is used. For example, in Table 1, the position of Edge 1 is then noted plus a surface distance/width to Edge 2. Edge 2 thereby depends on the position of Edge 1. Similar to the examples in Figure 4B,C, a height is noted positive and a depth is negative. The slope notation is identical as well.

It is chosen to run all parameters with 10 different values for each parameter. Having five parameters with 10 values each gives 100,000 cases. Presentation of a five-dimensional simulation matrix is a significant effort and not practical; hence, a series of simulations are made for two parameters at a time, yielding 100 sets of polars for each set of simulations. The selected inputs, combinations and results can be seen in Section 4.

It is important to notice that the edges can be on the same side of the airfoil. Figure 4 is illustrating cavities where Edge 1 is on the pressure side (p.s.) and Edge 2 on the suction side (s.s.). Some of the simulations is made where both edges are on either the pressure or suction side, illustrating, for example, a smashed bug on the blade or a small delamination from a transport damage.
SIMULATIONS

The simulation results are obtained by usage of CFD, which consist of two major parts, the grid and the numerical setup, which is described in Sections 3.1 and 3.2. The output from the simulation used in the results is described in Section 3.3.

3.1 Grid

The NACA 633-418 airfoil found the basis for the grid used in the simulation. The simulations are validated in previous work, the choice of airfoil used for simulation of an airfoil with a smooth surface.

As mentioned in Section 2, this article contains several hundred cases. Each case has an individual grid based on the same properties. Each grid has 512 cells in the circumferential direction and 384 cells in the normal direction.

TABLE 1 Input values for variation in cavity position and width on the pressure side

<table>
<thead>
<tr>
<th>Figure 11 Total: 100</th>
<th>Edge 1 Position</th>
<th>Edge 2 Position (Width)</th>
<th>Edge 1 Slope</th>
<th>Edge 2 Slope</th>
<th>Depth (→) Height (+)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1%c p.s. Edge 1 + (0.15%c)</td>
<td>85°</td>
<td>85°</td>
<td>–0.1 %c</td>
<td></td>
</tr>
<tr>
<td></td>
<td>2%c p.s. Edge 1 + (0.20%c)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>10%c p.s. Edge 1 + (0.60%c)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

FIGURE 4 (A) Shape of the NACA 633-418 airfoil with a zoom area marked. (B) The marked zoom area from (A) with marks of Edges 1 and 2. (C) Zoom on Edge 1 showing the smooth airfoil and 4 different cavities. (D) Zoom on a cavity with Edge 2 described as Edge 1 plus a width measured on the surface [Colour figure can be viewed at wileyonlinelibrary.com]
The distribution of the 512 cells are done with an algorithm implemented in Matlab, satisfying a set of constraints, such as maximum difference in cell length of neighboring cells of 10%, minimum number of cells in cavities depending on cavity dimensions, and a higher concentration of cells in location of high pressure gradients.

The distribution of the 384 cells follows a hyperbolic tangent distribution function. The wall cells have a height of \( \approx 1 \cdot 10^{-6} \) of the chord length. This results in a \( \gamma^+ \) value of \( \approx 0.1 \)–0.2. The cells most distant from the airfoil have a height of three-chord lengths.

The grid is constructed using the DTU Wind in-house HypGrid2D grid generator\(^2\). It is set to generate an o-mesh with a radius of 45 times the chord length. The outlet is set to cover around 40\(^\circ\) to 50\(^\circ\) on both sides of the trailing edge. If a case requires a lot of points in the cavities, the total distribution is changed slightly and might change the outlet size. The grid is divided into blocks to allow parallel processing. The domain, blocks, inlet, and outlet are illustrated in Figure 5.

Figure 6 shows a close up of the airfoil. Only a coarser grid with every other gridline in both directions is shown. A higher concentration can be seen on the leading edge, the disturbances, the trailing edge, and close to the wall.

Figure 7 shows an example of Edge 2, in full grid resolution, located in 6% chord and with a depth of 0.6%c. The edge is in this case defined by 12 cells. The major parameter for distributing the number of cells is a scale of the 2.25th root of the curve length. As an example, a depth of 0.3%c will give eight cells and 0.1%c will give five cells.

### 3.2 Numerical setup

The CFD solver used in this article is the in-house DTU software EllipSys2D. EllipSys2D is an incompressible finite volume RANS flow solver, which in this work uses the SIMPLE algorithm to solve the Navier–Stokes equations.\(^1\)\(^9\)\(^,\)\(^2\)\(^0\)\(^,\)\(^2\)\(^3\) The QUICK scheme is used to discretize the convective terms; see Leonard.\(^2\)\(^4\) A relaxation of 0.5 is used on all velocities and a relaxation of 0.15 on the pressure in all simulations. For convenience, a dimensionless study has been made. Therefore, the density of the fluid is 1 kg/m\(^3\), the chord is 1 m, and the dynamic viscosity is \(3.333 \cdot 10^{-7}\)
m²/s. The inlet is laminar flow with a velocity of 1 in the x-direction for \( \alpha = 0 \) for an Angle of Attack (AoA) of 0°. These inputs result in a Reynolds number of three million. All simulations run until convergence with the limit of \( 1 \cdot 10^{-6} \) of the residuals from the initial conditions.

The simulation runs on three different grid levels, where the first two are on a coarser grid and eventually the final result on the finest grid. The results from the coarser grid are used as initial conditions in the finer grid. The grid is divided into 48 blocks; hence, 1 node with 16 cores is allocated for every AoA.

Menter’s \( k-\omega \) with SST extension is used as turbulence model. The eN model by Drela–Giles is used to simulate transition of the flow. The critical N factor is specified as 9, and a relaxation factor of 0.5 is applied to the model. The system is heavily relaxed compared to standard computations, as the disturbances often is seen to trigger the transition point resulting in more flow dynamics. This could be solved by using an unsteady solver; however, to keep calculation time low, higher relaxation is used with a steady solver.

The assumption of a steady solution holds in most cases; however, some cases showed behavior as an unsteady solution, not being able to converge despite changing solver parameters and cell distribution. Three AoAs have been discarded from the plots due to this and substituted with one of the neighboring cases results. These are marked with an X and only occurs in Figure 14. However, the governing tendencies, behavior, and conclusions remain unaffected.

### 3.3 Output

A typical method for analyzing the aerodynamics of an airfoil is to investigate the lift and drag coefficients for 20° around zero lift, which is a typical no-stall range. Using that method with this work would produce a vast amount of curves, making it impossible to analyze.

Wind turbines operate in various ways, but a typical rule of thumb is to design the controller such that the blade operates at an AoA corresponding to design lift (often around 0.7 to 0.8 times the maximum lift), which often is close to the maximum lift/drag. This is close to 8° AoA for the smooth airfoil. A blade with LER does not usually change pitch scheme, and since maximum lift becomes lower with LER, the blade will operate closer to maximum lift. It is chosen to plot results for 6°, 8°, and 10° AoA based on these considerations. However, it should be noted that it is not straightforward whether it should be chosen to analyze for constant AoA or for constant lift, because LER will change the induction field around the rotor. With the lower lift caused by the LER, both the lift value and the AoA will in general change.

Figure 8A,B shows an example of lift and drag for the smooth simulation and a simulation with LER. The 6°, 8°, and 10° AoA are marked for clarity.

Plotting only three AoAs reduces the amount of data. In order to compare the three AoAs, the change in lift and drag are all normalized w.r.t. the smooth simulation. This is shown in Figure 8C,D. It can be seen that the changes differs a lot when moving to lower or higher AoAs;

![Figure 8](attachment:figure8.png)
however, it was chosen to omit these as a wind turbine rarely operates at these AoA's for wind speeds below rated power when maximizing the power coefficient. The effect of the $e^{n}$ model described in Section 3.2 can be seen illustrated in Figure 9. The LER case used in the example has a cavity stretching from 0.01 x/c on the pressure side to 0.01 x/c on the suction side. The transition is located on the edge of the cavity on the suction side for all AoAs. When the stagnation point moves to the right of the cavity edge, the transition point on the pressure side closely follows the one of the smooth airfoil. Simply substituting the edges with a forced transition will not have the same effect. As seen in the contour plots in Section 4, the height of a step has a significant impact on both lift and drag, whereas a forced transition point would have a fixed change in lift and drag. The minimum depth of a step that triggers transition depends on a series of variables such as the boundary layer thickness and local Reynolds number; see Bak et al.\textsuperscript{11}

All plots contain a normalized lift decrease and normalized lift/drag decrease. It is chosen to plot lift/drag instead of drag, because the lift/drag ratio directly affects the rotor performance. Furthermore, it is possible to have a similar range on the plots, as well as staying in “decreases,” as drag is seen to increase. An increase in drag will result in a decrease in normalized lift/drag. The plot chosen is a 2D color map, as it allows for two parameters to change. Each plot has a related table and text describing the input and results. All figures contains 100 results, that is, 300 lift and 300 drag values in total for each plot. The colorbar is fixed in all plots, ranging from 0% to 50% for normalized lift decrease and 0% to 100% for normalized lift/drag decrease. The fixed values makes it easier to compare plots with different inputs. White is minimum, going into yellow, hence red, ending with black as maximum. No plot has a value above the colorbar range. The increments are variable and set by the difference between minimum and maximum values in a plot.

As mention, a verification of the method has been performed in a previous study by the same authors, and for ease of the reader, one of the figures is repeated in Figure 10, where one can see the correspondence between wind tunnel data of an airfoil with zigzag tape compared to CFD simulations made in a similar setup. For further details, see Kruse et al.\textsuperscript{21} The setup has a 0.4-mm height zigzag tape on the suction side at 0.02 x/c.

**FIGURE 9**  Left: Transition location on the suction and pressure side of a smooth profile and a profile with LER. Right: Profile with LER as a simple cavity stretching to x/c = 0.01 on the suction and pressure side [Colour figure can be viewed at wileyonlinelibrary.com]

**FIGURE 10**  Comparison of lift and lift/drag ratios for simulations and WT test with 0.4-mm ZZ tape\textsuperscript{21} [Colour figure can be viewed at wileyonlinelibrary.com]
In Figure 10, "WT zz 0.4" refers to the wind tunnel experimental data of the airfoil with 0.4-mm zigzag. "CFD 2D zz 0.4" refers to simulation done in 2D, where the zigzag tape can be seen as a simple protuberance. "CFD 3D zz 0.4" is a fully resolved airfoil with zigzag tape. The spanwise boundary conditions are cyclic, and hence, the tunnel walls and the effect from these are not represented. Lastly, the "CFD pseudo-2D zz 0.4" is an extruded 2D geometry, making it a 3D airfoil with a strip (rather than zigzag tape). The lift and drag values are scaled as the difference between the rough and the clean values scaled with the clean values. The results show an overall good agreement for all three simulations at AoA up to 8°, where it starts to deviate. Another observation is that the higher the zigzag tape is, the more accurate is the 3D simulation. The accuracy of the simulations was deemed acceptable for the purpose of the parametric study in this paper.

4 | RESULTS

This section contains five sets of results, each set with two variables.

4.1 | Cavities with widths up to 0.6%c at the pressure side

An example of how some variables affect the airfoil performance is illustrated by one common example, seen in Figure 11. The input can be seen in Table 1. The first set of results illustrated in Figure 11 shows below 1% change in lift for all three AoAs. The lift/drag decreases around 12% for 6° AoA and less for higher angles. Due to the high pressure on the pressure side of the airfoil, the disturbance has little effect despite changes in chordwise position and width. There is a small but noticeable plateau change on both the lift and lift/drag, when the disturbance is downstream of the stagnation point.

Surface imperfections that result in minor changes in performance are not unique. A bump with the same Edges 1 and 2 position gave similar results. All simulations have been carried out with 85° slope on both edges. A study was made on the edge angles as well, ranging from 45° to 90° with increments of 5°. It was done on different types of cavities and bumps. Common for all was that the slope had little effect on the results, similar to the results shown in Figure 11. The maximum variation of the Edge 1 slope was ±0.5% in lift decrease and ±1% in lift/drag decrease compared to the surface without cavities and bumps. For the Edge 2 slope, the variation was ±1% in lift decrease in ±2.5% in lift/drag decrease compared to the surface without cavities and bumps. A shallow slope, in general, did result in less drag; however, the results were all dominated by the depth and position.

It was chosen to simulate all other cases with slopes of 85° for consistency as well as good convergence of simulation for most results.

![Figure 11](Colour figure can be viewed at wileyonlinelibrary.com)
Presentation of results with minor changes, being Edges 1 and 2 slopes, have been omitted in this paper to focus on parameters that influences the aerodynamic performance the most. It was chosen to include them in the tables because it is describing the geometry and to inform that they have been analyzed.

4.2 | Cavities with widths up to 0.6%c at the suction side

Figure 12 shows results of inputs similar to the inputs of the results in Figure 11. The difference is that the cavity is placed on the suction side of the airfoil instead of the pressure side. The inputs can be seen in Table 2.

A cavity located on the suction side results in a drop of around 5% in lift for 6° AoA and a lower drop for higher AoAs. The lift/drag decreases more aggressively, resulting in a maximum change in lift/drag of 40% at 6° AoA.

The changes are sensitive to both the position and the width (surface distance) of the cavity. The worst situation is it when the cavity is located close to the leading edge and is very wide.

4.3 | Bumps with widths up to 0.6%c at the suction side

Figure 13 shows results for a bump with the same locations and dimensions as in Figure 12. The inputs can be seen in Table 3.

The result from introducing the dump is shown in Figure 13 and shows a higher lift penalty compared to voids of the same dimensions as shown in Figure 12. The cavity had less effect at higher AoAs, where the bump results in up to 12% lift reduction for 10° AoA. The highest aerodynamic influence can be seen for the same Edge 2 position for all distances. For the 6° AoA, the Edge 2 position of 4%c is the worst, for 8° AoA, 10%c is the worst.
it is $3\%c$, and for $10^\circ/C14$ AoA, it can be found at $2\%c$. These positions are where Edge 2 is perpendicular to the flow direction, resulting in the highest form drag.

4.4 | Wide cavities covering the leading edge

The next step is to move to situations where the cavity covers the leading edge as in the example seen in Figures 1−4. The positions of Edges 1 and 2 are varied in combinations described in Table 4 with the results plotted in Figure 14.

Significant reductions in lift and lift/drag are seen with up to 10% loss in lift and 40% loss in lift/drag. However, for AoA $6^\circ$ and $8^\circ$, the decrease in lift and lift/drag seems to be quite constant despite of the position of Edges 1 and 2 for $6^\circ$ and $8^\circ$ AoA. The drag is lower when Edge

---

**FIGURE 13** Results for variation in bump position and width on the suction side [Colour figure can be viewed at wileyonlinelibrary.com]

**TABLE 3** Input values for variation in bump position and width on the suction side

<table>
<thead>
<tr>
<th>Figure 13 Total: 100</th>
<th>Edge 1 Position</th>
<th>Edge 2 Position</th>
<th>Edge 1 Slope</th>
<th>Edge 2 Slope</th>
<th>Depth (−) Height (+)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1%c s.s.</td>
<td>Edge 1 + (0.15%c)</td>
<td>85$^\circ$</td>
<td>85$^\circ$</td>
<td>0.1%c</td>
<td></td>
</tr>
<tr>
<td>2%c s.s.</td>
<td>Edge 1 + (0.20%c)</td>
<td>...</td>
<td>...</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10%c s.s.</td>
<td>Edge 1 + (0.60%c)</td>
<td>...</td>
<td>...</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**TABLE 4** Input values for variation in leading edge cavity position for Edges 1 and 2

<table>
<thead>
<tr>
<th>Figure 14 Total: 100</th>
<th>Edge 1 Position</th>
<th>Edge 2 Position</th>
<th>Edge 1 Slope</th>
<th>Edge 2 Slope</th>
<th>Depth (−) Height (+)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5%c p.s.</td>
<td>0.5%c s.s.</td>
<td>85$^\circ$</td>
<td>85$^\circ$</td>
<td>−0.1%c</td>
<td></td>
</tr>
<tr>
<td>1.0%c p.s.</td>
<td>1.0%c s.s.</td>
<td>...</td>
<td>...</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.0%c p.s.</td>
<td>5.0%c s.s.</td>
<td>...</td>
<td>...</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

it is $3\%c$, and for $10^\circ$ AoA, it can be found at $2\%c$. These positions are where Edge 2 is perpendicular to the flow direction, resulting in the highest form drag.
1 is positioned upstream of the stagnation point. For 10° AoA, the penalty is high when Edge 1 is located at the stagnation point. All three AoAs show that the penalty is higher, the closer Edge 2 is to the leading edge. Note that the three values marked with X have been omitted due to an unsatisfied convergence criteria.

4.5 | Wide symmetric cavities with variations of depth covering the leading edge

The last parameter that has been investigated is the depth/height. A LER with a height of 1%c starts to look more like ice accretion, which is outside the scope of this work. Based on this, it was chosen to show the results for depth of LER as one parameter and a combination of Edges 1 and 2 position as the other parameter. Table 5 shows the inputs with the results plotted in Figure 15. Note that Edges 1 and 2 have the same chordwise positions, there is no cross combination of these.

The aerodynamic properties for LER seem to depend strongly on the depth of the LER for all three AoAs. The lift drops between 5% for 0.1%c depth and up to 35% for 1%c depth. The lift/drag is strongly degraded when the depth exceeds 0.3–0.4%c. A reduction of up to 90% in lift/drag at 12° AoA makes the depth the most critical parameter in this study. A stronger correspondence between edge positions and lift reduction is seen when the depth is increased. Having Edge 2 close to the leading edge is much worse than having the LER extended back to 10%c.

**TABLE 5** Input values for variation in leading edge cavity position for Edges 1 and 2 combined with a depth change

<table>
<thead>
<tr>
<th>Figure 15 Total: 100</th>
<th>Edge 1 Position</th>
<th>Edge 2 Position</th>
<th>Edge 1 Slope</th>
<th>Edge 2 Slope</th>
<th>Depth (–) Height (+)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1%c p.s.</td>
<td>1%c s.s.</td>
<td>85°</td>
<td>85°</td>
<td>−0.1%c</td>
</tr>
<tr>
<td></td>
<td>2%c p.s.</td>
<td>2%c s.s.</td>
<td>85°</td>
<td>85°</td>
<td>−0.2%c</td>
</tr>
<tr>
<td></td>
<td>..</td>
<td>..</td>
<td>..</td>
<td>..</td>
<td>..</td>
</tr>
<tr>
<td></td>
<td>10%c p.s.</td>
<td>10%c s.s.</td>
<td>..</td>
<td>..</td>
<td>−1%c</td>
</tr>
</tbody>
</table>

**FIGURE 14** Results for variation in leading edge cavity position for Edges 1 and 2 [Colour figure can be viewed at wileyonlinelibrary.com]
CONCLUSION

This parametric study was conducted on simplified disturbances on a NACA 633-418 airfoil, at a Reynolds number of three million. The simplified disturbance was described by two steps forming either a bump or a cavity. The major shape descriptors was position of two edges, the angle of these and the depth or height of the disturbance. In general, the results depends on whether the disturbance is constructed as covering the leading edge or as a disturbance on either the suction or pressure side. The conclusion sums up the governing tendencies.

1. The slopes within a range of 45° to 90° of Edge 1, that is, the edge first encountered when moving clockwise around the airfoil from the trailing edge, had negligible impact on the aerodynamic performance of the airfoil of less than ±0.5% on lift and ±1% on lift/drag.
2. The slopes within a range of 45° to 90° of Edge 2, that is, the edge first encountered when moving counter clockwise around the airfoil from the trailing edge, had an impact on the aerodynamic performance of the airfoil of less than ±1% on lift and ±2.5% on lift/drag.
3. The position of Edge 1 was tested in various positions in the frontal 10%c of the airfoil. When Edge 1 was placed on the suction side of the stagnation point, the highest loss were observed, especially when the position were very close to the stagnation point. With the Edge 1 downstream of the stagnation point on the pressure side lower losses were observed. The position of Edge 1 has the largest impact on performance in low AoAs. The variation in Edge 1 position may account for up to 10% on lift and 20% on lift/drag.
4. The position of Edge 2 follows the structure of Edge 1 positions. An Edge 2 location close to leading edge gives the most performance penalty. If Edges 1 and 2 are both located upstream of the stagnation point, the location of Edge 2 has less impact when simulating a bump. The variation in Edge 2 position may account for up to 5% on lift and 10% on lift/drag, highly depending on Edge 1 location.
5. The depth/height parameter was tested for a range of 0.1–1%c. Changes in bump height has severe impact on the performance. A leading edge with added material is out of the scope for this study, as it would be closer to an ice accretion analysis. Analysis of missing material on the leading edge (cavities) shows that the depth plays a significant role. Even with the same location of Edges 1 and 2, the depth in the specified range is seen to account for up to 30% on lift and 80% on lift/drag.

The governing conclusion is that the most important parameters to estimate or measure when analyzing the performance loss of a NACA 633-418 airfoil is the position/extension and depth/height of a disturbance whereas a detailed shape of the disturbance is of less importance.

ACKNOWLEDGEMENTS

The authors want to acknowledge the Danish Energy Agency for the support and funding of this project, the "Leading Edge Roughness on Wind Turbine Blades," 64015-0046. The acknowledgement also goes to the Technical University of Denmark and Power Curve ApS.

FIGURE 15  Results for variation in in leading edge cavity position for Edges 1 and 2 and combined with a depth change [Colour figure can be viewed at wileyonlinelibrary.com]
REFERENCES


How to cite this article: Krog Kruse E, Sørensen NN, Bak C. A two-dimensional quantitative parametric investigation of simplified surface imperfections on the aerodynamic characteristics of a NACA 632-418 airfoil. Wind Energy. 2021;24:310–322. https://doi.org/10.1002/we.2573